

2012 International Workshop on Information and Electronics Engineering (IWIEE)

## Numerical Simulation in Influence of Forward-Facing Cavity on Aerodynamic Heating of Hypersonic Vehicle

Haibo Lu<sup>a\*</sup>, Weiqiang Liu<sup>a</sup>

<sup>a</sup>*College of Aerospace and Material Engineering, National University of Defense Technology, Changsha ,410073, China*

---

### Abstract

A CFD study on the effect of forward-facing cavity upon aerodynamic heating on the hypersonic vehicle nose is conducted, by means of which the flow field parameters, heat flux distribution along outer body surface are obtained. The numerical simulation result is validated by experiment and the cooling effect of the forward-facing cavity with different dimension is analyzed. The CFD results shows that the forward-facing cavity configuration dose well in cooling the nose of hypersonic vehicle especially at the stagnation point area. The deeper the cavity is, the smaller the heat flux is. The maximal heat flux along the outer body surface dose not locates at the peak of the sharp lip, but after it, and the length of the cavity (L) has little effect on the location of the maximal heat flux.

© 2011 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Harbin University of Science and Technology. Open access under [CC BY-NC-ND license](#).

**Keywords:** Forward-facing cavity; Thermal protection; Hypersonic vehicle; Aerodynamic heating; CFD

---

### 1. Introduction

There is a severe aerodynamic heating when an aircraft travels at high velocity. How to design the thermal protection system of these vehicles is always a focus for scholars in thermal protection fields.

A body containing a forward-facing cavity under a supersonic flow was introduced firstly by Hartmann [1] in 1921, which was used as a new technique for producing sound of high intensity and discrete frequency. In 1959, Burbank [2] reported this idea as a thermal protection technique for the nose-tip of hypersonic vehicles firstly, after that, more and more researchers study on it for its simple structure and excellent thermal protection effect [3~9].

---

\* Corresponding author. Tel.: +0-000-000-0000 ; fax: +0-000-000-0000 .

E-mail address: [lhbboo@sohu.com](mailto:lhbboo@sohu.com).

Yuceil [3] et al. using an infrared camera indicated that larger-diameter shallow cavities created a stable “cool ring” in the vicinity of the cavity lip, with temperatures locally lower than those of a simple spherical nose. W A Engblom and D B Goldstein[4] research on the distribution of the heat flux and pressure along nose-tip with forward-facing cavity, which  $L/D$  is 0.75 and 2.(Fig.1). Siltond [5] et al. study on the the effects of the cavity on ablation onset time and they validated the laminar assumption in CFD. Later, they investigated how to reduce the severe heating and delay the ablation onset [6, 7]. In addition, an experimental parameter study is un-dertaken to optimize the forward-facing cavity geometry for the most delayed ablation onset. The parameters of cavity length, lip radius, and diameter are independently optimized for a given nose-tip diameter. They found that the best  $L$ , for a given  $D_n$ , was four times  $D$ . The best  $r$  was one-fourth the difference between  $D_n$  and  $D$ . Experimental results (Ma 4.5) of the nose-tip heating with a forward-facing cavity ,has published by F Seiler [8]. S Saravanan [9] et al. investigated the effects of a forward-facing cavity on heat transfer and aerodynamic coefficients. Numerical simulation was carried out with steady-state flow assumption and had a good agreement with their tests in hypersonic shock tunnel HST2, at hypersonic Mach number of 8.

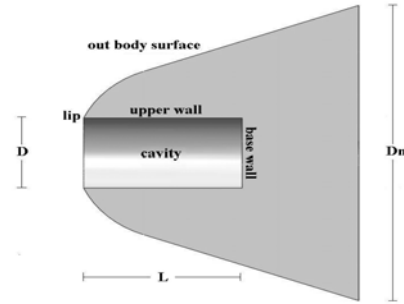


Fig.1 Nose-tip with forward-facing cavity

### Nomenclature

$C_{p\infty}$	was the constant-pressure specific heat of the freestream,
$D$	diameter of the cavity
$D_n$	diameter of the bottom of the nose-tip
$L$	depth of cavity
$Ma_\infty$	Mach number of freestream.
$Pr$	Prandtl number
$P_\infty$	Free stream static pressure
$q_w$	heat flux
$St$	Stanton Number
$St_{max}$	the maximum of the Stanton number
$T_\infty$	Free stream static temperature
$T_{aw}$	temperature of the thermal isolation wall
$T_w$	Wall temperature
$u_\infty$	velocity of the freestream
$x$	axis of bank of the nose-tip
$\rho_\infty$	density of the freestream
$\gamma$	specific heat ratio

## 2. Numerical Method

In the present study, the 3-D Navier-Stokes equations are used as governing equations. The convective terms are approximated using AUSM-DV splitting method, a MUSCL approach with Min-mod limiter is implemented to increase the numerical accuracy [10], and central difference method for the viscous terms. The LU-SSOR scheme is used for the time integration.

The boundary condition is that  $Ma_\infty=8$ ,  $P_\infty=205\text{Pa}$ ,  $T_\infty=143\text{K}$ . The wall boundary condition is used at the nose-tip surfaces and the fluid at these surfaces is assumed to have a no-slip condition, and the wall temperature is assumed isothermal ( $T_w=300\text{K}$ ). In order to get stable simulation results, a steady-state flow condition is assumed. The grid of simulation model on the symmetry plane and on the wall of the nose-tip is shown in Fig. 2 and Fig.3.

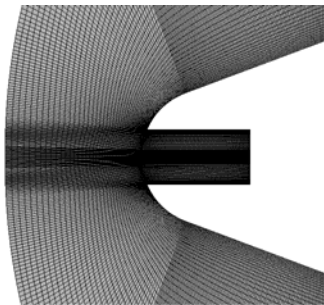


Fig.2 Symmetry plane grid (L=24mm,D=12mm)

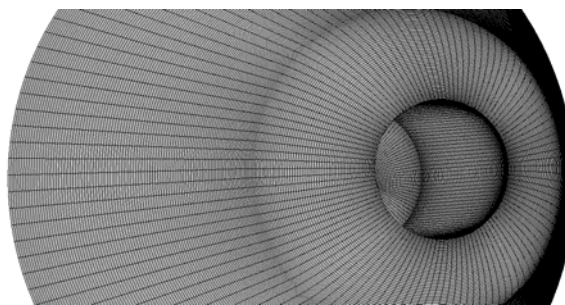


Fig.3 Grid on the object surface of nose-tip (L=24mm,D=12mm)

## 3. Numerical and Experimental Results Comparison

In order to compare with the experimental results [9], the boundary conditions used in the simulation are based on the experimental free stream condition. Three validating examples (blunt cone, nose-tip with L 24mm, D 6mm and 12mm cavity) are established here same as the experimental model. The Stanton number along the outer body surface of validating examples simulation and experiment from [9] are compared in Fig. 4.

A big agreement is shown between simulation and experiment results in these figures. Some errors come from the assumption of simulation model, counting error and experimental measurement. The distribution of Mach number and temperature on symmetry plane near nose-tip of the three validating examples is shown in Fig. 5, Fig. 6. Comparing these figures, the use of forward-facing cavity dose not change the shape of the bow shock but have a great effect on the distribution of temperature behind the shock, especially near the stagnation area.

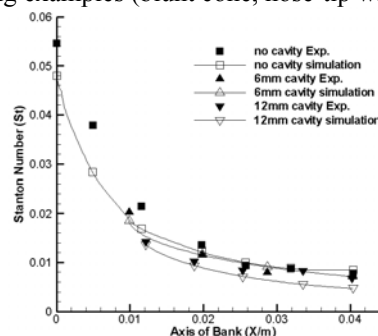


Fig.4 Stanton number along outer body surface comparison between CFD and experiment [9]

## 4. Results and Discussion

In order to investigate the influence of the forward-facing cavity on aerodynamic heating, as shown in Tab.1, calculation cases with different geometry of cavity were established.

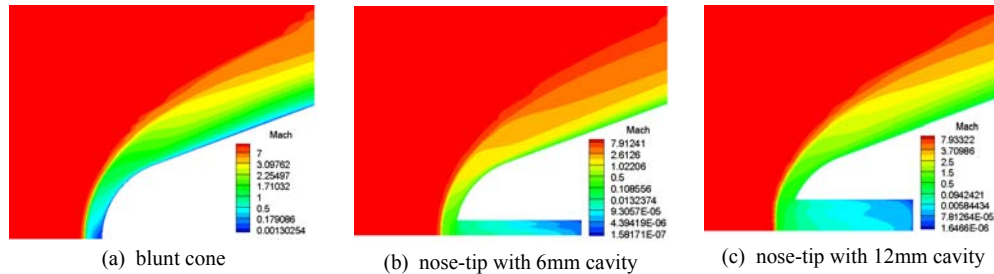


Fig.5 distribution of Mach number

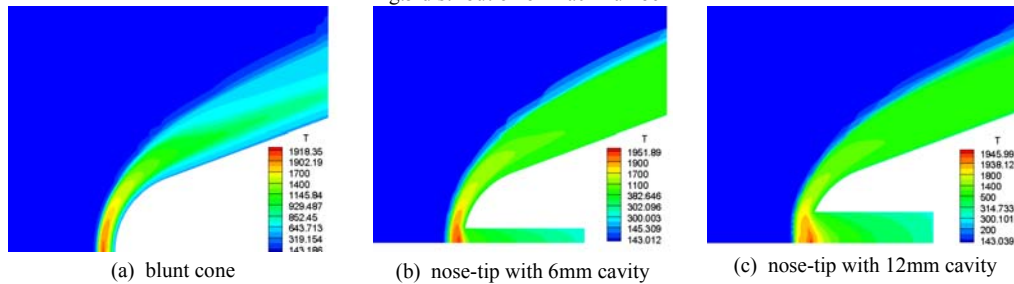


Fig.6 distribution of temperature/K

Table 1. Calculation cases of cavity with different geometry

Cases	L/D	L(mm)	D(mm)	Cases	L/D	L(mm)	D(mm)
1	1	12	12	3	2	24	12
2	1.5	18	12	4	2.5	30	12

Stanton number based on the freestream condition, is given by the expression

$$St = \frac{q_w}{(T_{aw} - T_w) \rho_\infty c_{p\infty} u_\infty} \quad (1)$$

$$T_{aw} = T_\infty \left\{ 1 + \sqrt[3]{Pr} \left[ (\gamma - 1) / 2 \right] M_\infty^2 \right\} \quad (2)$$

The Stanton number distribution along outer body surface for all cases is shown in Fig.7. As seen in Fig.7 (a), the forward-facing cavity configuration dose well in cooling the nose tip especially at the stagnation point area. The Stanton number of nose-tip with a blunt cone is 0.047998, but the nose-tip with cavity, the Stanton number on the lip is approximate to 0.026. The mean heat flux increases along the outer body surface to reach a peak value near the sharp lip, and then decreases fastly. The maximum heat flux and the location of it is given in Tab.2. It can be seen from the table that the deeper the cavity is, the smaller the maximum heat flux is and the length of cavity (L) has little effect on the location of the maximum heat flux. It is shown in Fig.7 (b,c,d) that not only the maximum heat flux but also whole heat flux along the nose have the trend that the deeper the cavity is, the smaller the heat flux is.

Table 2. The maximum heat flux and the location

case	1	2	3	4
St <sub>max</sub>	0.026298	0.026288	0.026155	0.026055
x/(mm)	1.448	1.4948	1.4948	1.5445

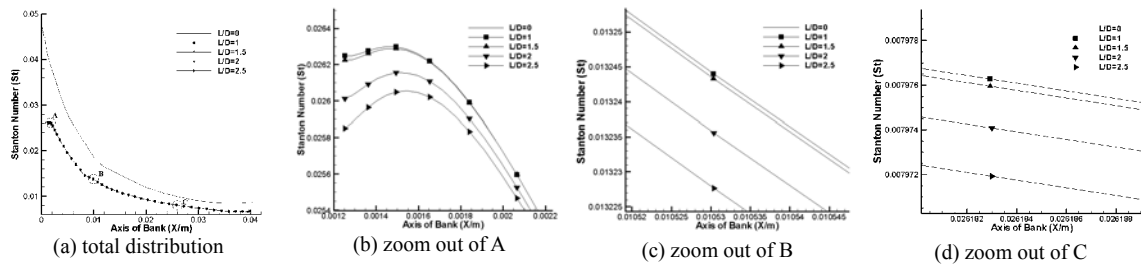


Fig.7 The Stanton number distribution along outer body surface for all cases ( $L/D=0$  is the blunt cone with no cavity)

## 5. Conclusion

- The forward-facing cavity configuration dose well in cooling the nose of hypersonic vehicle especially at the stagnation point area.
- The mean heat flux increases along the outer body surface to reach a peak value near the sharp lip, and then decreases fastly. The downtrend become slow at the crossover area of the hemisphere and cone.
- The deeper the cavity is, the smaller the heat flux along the outer body surface is.
- The maximal heat flux along the outer body surface dose not locates at the peak of the sharp lip, but after it, and the length of the cavity ( $L$ ) has little effect on the location of the maximal heat flux.

## Acknowledgements

The authors would like to thank for support of this work by Major Program of National Natural Science Foundation of China (90916018) and Research Fund for the Doctoral Program of Higher Education of China (200899980006).

## References

- [1] Hartmann J, Troll B. On a New Method for the Generation of Sound Waves. *Phys. Rev.* 1922; **20**: 719–727.
- [2] Burbank P B, Stallings R L. Heat-Transfer and Pressure Measurements on a Flat Nose Cylinder at a Mach Number Range of 2.49 to 4.44, NASA TM X-221, 1959.
- [3] Yuceil B, Dolling D S, Wilson D, A Preliminary Investigation of the Helmholtz Resonator Concept for Heat Flux Reduction AIAA 1993-2742, 1993.
- [4] W A Engblom, D B Goldstein. Nose-Tip Surface Heat Reduction Mechanism. AIAA 96-0354, 1996.
- [5] Siltan S I, Goldstein D B. Modeling of Nose Tip Ablation Onset in Unsteady Hypersonic Flow. AIAA 2000-0204, 2000.
- [6] Siltan S I, Goldstein D B. Ablation Onset in Unsteady Hypersonic Flow About Nose Tip with Cavity. *J. Thermophys. Heat Trans.* 2000; **14**: 421–434.
- [7] Siltan S I, Goldstein D B. Use of an Axial Nose-Tip Cavity for Delaying Ablation Onset in Hypersonic Flow. *J. Fluid Mech.* 2005; **528**: 297–321.
- [8] Sidra I Siltan, David B Goldstein. Optimization of An Axial Nose-tip Cavity for Delaying Ablation Onset in Hypersonic Flow. AIAA 2003-152, 2003.
- [9] S Saravanan, G Jagadeesh, K P J Reddy. Investigation of Missile-Shaped Body with Forward-Facing Cavity at Mach 8. *J. Spacecraft Rockets.* 2009; **46**: 557-591.
- [10] Azevedo J L F, Heidi K. Comparison of unstructured grid finite volume methods for cold gas hypersonic flow simulations. AIAA-98-2629, 1998.